

```

/*-----* C++ *-----*/
| ===== |
| \\ / F i e l d | OpenFOAM: The Open Source CFD Toolbox
| \\ / O p e r a t i o n | Version: 4.1
| \\ / A n d | Web: www.OpenFOAM.org
| \\ / M a n i p u l a t i o n |
/*-----*/
FoamFile
{
  version      2.0;
  format       ascii;
  class        volVectorField;
  object       U;
}
// ***** //

dimensions      [0 1 -1 0 0 0 0];

internalField   uniform (1.463 0 0);

boundaryField
{
  inlet
  {
    type          freestream;
    freestreamValue uniform (1.463 0 0);
  }

  outlet
  {
    type          freestream;
    freestreamValue uniform (1.463 0 0);
  }

  airfoil
  {
    type          fixedValue;
    value uniform (0 0 0)
  }

  frontAndBack
  {
    type          empty;
  }
}
// ***** //

```